Comparison Between Surface Heating and Volumetric Heating Methods inside CANDU Reactor Moderator Test Facility (MTF) Using 3D Numerical Simulation

Araz Sarchami*1, Nasser Ashgriz2, Marc Kwee3

Mechanical and Industrial Engineering Dept., University of Toronto
5 King's College Road, M5S 3G8, Toronto, Canada

*1sarchami@mie.utoronto.ca; 2ashgriz@mie.utoronto.ca; 3marc.kwee@brucepower.com

Abstract

Three dimensional numerical simulations are conducted on the CANDU Moderator Test Facility (MTF). Heat generation is modelled through both surface heating employed in experimental setups and volumetric heating, the actual heating method in real reactor. The result shows that while the asymmetry in temperature distribution is visible in both cases, and the locations of hot and cold zones are similar, but the temperature range is different. Although the heat input to both cases is identical but the temperatures are better distributed in volumetric heating case since the heat source is more uniformly distributed in the liquid.

Keywords

CANDU; Moderator; Numerical Simulation; Buoyancy; Surface Heating; Volumetric Heating

Introduction

Canadian Deuterium Uranium (CANDU) nuclear reactor is a Pressurized Heavy Water Reactor (PHWR) using heavy water as moderator in a horizontal, cylindrical tank (the calandria tank). The CANDU power reactor is composed of few hundred horizontal fuel channels in a large cylindrical calandria vessel, each of which consists of an internal pressure tube (containing the fuel and the hot pressurized heavy water primary coolant), and an external calandria tube separated from the pressure tube by an insulating gas filled annulus. The calandria vessel contains cool low-pressure heavy-water moderator that surrounds each fuel channel.

Several experimental and numerical studies have been conducted to investigate thermal hydraulics

inside the moderator tank. Khartabil et al. conducted three-dimensional moderator circulation tests in the Moderator Test Facility (MTF) in the Chalk River Laboratories of Atomic Energy of Canada Limited (AECL). MTF, a ¼ scale of CANDU Calandria, with 480 heaters that simulate 480 fuel channels, is designed to study moderator circulation at scaled conditions that are representative of CANDU reactors.

Carlucci and Cheung investigated the two-dimensional flow of internally heated fluid in a circular vessel with two inlet nozzles at the sides and outlets at the bottom, and found that the flow pattern was determined by the combination of buoyancy and inertia forces. Austman et al. measured the moderator temperature by inserting thermocouples. Huget et al. and conducted 2-dimensional moderator circulation tests at a 1/4-scaled facility. Sion measured the temperature profile of the D2O moderator inside a CANDU reactor, within the calandria vessel, by means of a specially instrumented probe introduced within the core. The results have established the feasibility of in-core moderator temperature measurement.

Hohneet.al. studied the influence of density differences on the mixing of a pressurized water reactor. A transition matrix from momentum to buoyancy-driven flow experiments was selected for validation of the computational fluid dynamics software ANSYS CFX. The results of the experiments and the numerical calculations show that mixing strongly depends on buoyancy effects.

Yoon et. al used a computational fluid dynamics model to predict moderator circulation inside the CANDU reactor vessel. The buoyancy effect induced by the internal heating was accounted for by the Boussinesq approximation. The governing equations were solved by CFX 4. They did a parametric analysis

and since their simulation was steady state, it was a base for future transient simulations. In their next paper, Yoon et. al. developed another computational fluid dynamics model by using a coupled solver and did the simulation for Wolsong Units 2/3/4.

Heat generation inside the tank has been modelled using two different methods: volumetric heating method, and surface heating method. Volumetric heating is the actual heating method in real reactor, which occurs through fission heat generation and gamma rays from fission products. But since application of this method in experimental study is not feasible, another method which is called surface heating is used, in which, heated rods are used inside the tank to heat the water. In this study, both methods are considered in separate simulations in order to compare the results and study the effect of heating method variation on temperature and velocity distribution inside the tank.

Problem Setup

MTF Geometry

The MTF tank, a ¼ scale of Bruce B Calandria tank, comprises a cylindrical tank with eight inlet nozzles (four at each side tank) and two pipes as outlets at the bottom of tank as shown in Fig 1 and Fig 2. The MTF tank consists of 2115 mm diameter cylindrical tank with 1486 mm length. The outlet pipes are 152 mm in diameter and there are total of 480 tubes with diameter of 33mm inside the tank.

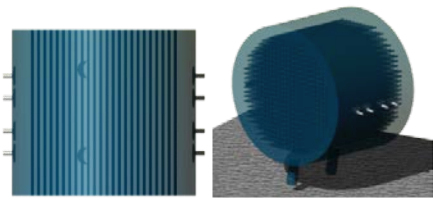

FIG. 1 MTF TANK GEOMETRY

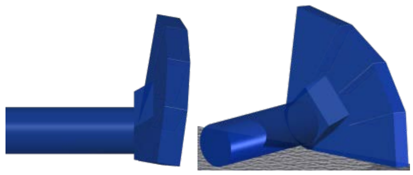

FIG. 2MTF NOZZLE GEOMETRY

Operating Conditions

During the normal operation of CANDU reactor, the cold moderator water enters the tank through eight nozzles, four nozzles at each side, and heated fluid exits from two outlet pipes at the bottom of the tank. Throughout the operation, two major flow characteristics are identified inside the tank: buoyancy driven fluid flows formed by the internal heating, and momentum driven fluid flows by the jet flows through the inlet nozzles, respectively. The flow behaviour depends on the operating conditions, such as, moderator mass flow rate and its temperature, and the rate of heat influx to the moderator. The operating conditions for the MTF used in the simulation are listed in Table 1.

TABLE 1 MTF OPERATING CONDITIONS

NOMINAL CONDITIONS	MTF
Power (kW)	1100
Mass flow rate (kg/s)	23
Number of nozzles	8
Number of outlets	2

Numerical Methodology

Fluent V12 is used as the numerical code for the simulation. Three dimensional geometry of the case is considered. The simulation is performed using unsteady, 2nd order implicit solver. RNG k-epsilon model with wall functions is used for turbulence modelling as the flow inside the tank is fully turbulent. Buoyancy effects are accounted for in the fluid density calculation method and the energy equation is solved for heat transfer inside the tank.

Surface heating method is modelled through heat influx at the boundaries of the tubes inside the calandria. Since the heat flux inside the actual tank is dependent on the coordinate along the length of the tank, the heat influx in numerical model is divided into 12 zones along the tank length and every zone has a different influx of heat at its boundary. Moreover, each zone is divided into inner and outer tank zone to model the heat transfer more accurately. The distribution of the heat loads is obtained from the actual operating data and experimental practices. The heat load is at its maximum toward the centre of the tank and reduces as it goes toward to ends. Volumetric heating is modelled using heat source for different sections of the tank.

Mesh Construction

An unstructured non-uniform tetrahedral and hexahedral mesh was used to construct meshes in the full MTF tank. A total of 3200000 meshes were generated using the commercial software Gambit, shown in Fig 3.

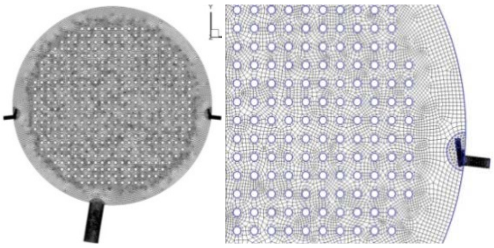

FIG. 3 MESH GENERATION

Results

Moderator tank is designed to receive a steady flow of heavy water as well as steady heat flux. Therefore, it is expected to achieve a steady state condition for flow and thermal distributions inside the tank. However, experimental observations have revealed that the thermal conditions inside the tank never reach a steady state, as evident by the measured temperature fluctuations. Three dimensional transient numerical simulations of the MTF tank with surface heating and volumetric heating show the same trend. As mentioned before, MTF tank is simulated using both surfaces heating and volumetric heating methods. In the actual reactor, the fluid is heated through nuclear radiation, resulting in a volumetric heating. In this case, the heat is distributed throughout the domain, rather than transferring only through the surfaces of the tubes. Therefore, the fluid in the tank is heated more uniformly and the tube surface temperatures are expected to be relatively lower than those in the surface heating case. Temperature and velocity distributions and fluctuations inside the MTF with volumetric heating are compared here with those of MTF surface heating. The comparison is done in several planes as shown in Fig 4. The results clearly show lower temperatures in the volumetric heating case, as well as more uniform temperature distribution. For instance, in plane S, the highest temperature is 63ºC for the surface heating case, whereas 57^{o}C for the volumetric heating case. Similarly, the highest temperature in plane B2 is around 63^{o}C for the surface heating case, whereas it is 55^{o}C for the volumetric heating case. Generally, the temperature variations inside the volumetrically heated tank are smaller than those in the surface heated tank. Consequently, volumetrically heated tank has lower buoyancy effects than those in the surface heated case. Increment in buoyancy results in increased segregation of hot and cold and less mixing efficiency. Therefore, the hot region in the surface heating is larger. The larger temperature variation in the surface heating is due to its relatively higher tube walls temperatures compared to those in the volumetric case.

Fig 5 presents the velocity contours at different planes for both the surface and volumetric heating cases. One obvious point in plane S is that the jet impingement point in the volumetric heating is more towards the top centre of the tank. As it was noted earlier that stronger buoyancy effects push the impingement point more towards one side. Since the buoyancy effects are less significant in the volumetric heating, the wall jet impingement point is more towards the top centre and flow is distributed more evenly. Most of the fluid in plane S have velocities under 0.4 m/s and in plane B2 have velocities under 0.2 m/s. The velocities in plane S are larger due to the combination of the fluid from two neighbouring jets.

Comparison has been made in Fig 6 on the temperature contours in planes D1, D2, and SY which are located in the XY plane. The uniformity in temperature as well as small temperature variation from point-to-point in the volumetric heating case is clear in all of the planes shown in the figure. The hot region for both cases is stretched along the length of the tank. The high temperatures in plane D1 for the volumetric heating are around 58^{o}C and the low temperatures are around 44^{o}C.

Velocities in the D1 plane range from 0.7 m/s to as low as 0.1 m/s. Velocity gradients are visible at both ends near the wall and diminish towards the centre of the tank. Due to higher velocity gradient at both ends of the tank, better mixing occurs in those areas resulting in a lower and more uniform temperature distribution. Whereas, in the middle of the tank, in which less mixing is observed, the temperatures are higher.

Fig 7 shows the contour plots for the vertical planes SX and E1. A saddle shape temperature contour is obtained in the surface heating case which is not as clear in the volumetric heating case. This again is due to the weaker buoyancy forces in the volumetric heating, which cannot push the hot fluid further up towards the upper walls in regions with lower incoming fluid velocities, namely regions closer to the end walls.

Summary & Conclusions

A three dimensional transient numerical simulation of the moderator tank is conducted. Two different heating methods are used to simulate the flow inside the tank: Surface heating method and volumetric heating method. The latter is the process which occurs

in reality and the former is the practical method used in experimental studies. The purpose of this study is to compare the temperature and velocity distribution inside the tank for both methods. The numerical simulations are performed on a 24-processor cluster using parallel version of the FLUENT 12. During the transient simulation, various locations inside the tank are monitored for their temperature and velocity. The temperature and velocity distributions in different planes are employed to analyze the thermo-hydraulics of the moderator tank for both heating methods.

In the real reactor, the fluid is heated through nuclear radiation, resulting in a volumetric heating. In this case, the heat is distributed throughout the domain, rather than transfer only through the surfaces of the tubes. Therefore, the fluid in the tank is heated more uniformly and the tube surface temperatures are relatively lower than those in the surface heating case.

Generally, the temperature variations inside the volumetrically heated tank are smaller than those in the surface heated tank. Consequently, volumetrically heated tank has lower buoyancy effects than those in the surface heated case. Increase in buoyancy results in increased segregation of hot and cold and less mixing efficiency. Therefore, the hot region in the surface heating is larger. The larger temperature variation in the surface heating is due to its relatively higher tube walls temperatures compared to those in the volumetric case.

Temperature and velocity fluctuations at various points inside the tank are monitored and the results indicates that although the temperature and velocity distribution inside the tank are different for two cases, but the fluctuations are observed in both cases and their general behaviour can be categorized as:

- I large amplitude temperature fluctuations are mainly at the boundaries between the hot and cold (e.g., outer boundaries of the bundle, regions close to the penetrating secondary jet flows);
- II low amplitude temperature fluctuations are mainly in the core of the tank with more uniform temperature distributions (e.g., central and lower parts of the tank);
- III high frequency fluctuations are in the regions with high velocities (e.g., top boundaries at the interface between the wall jet flows and the inner core flows, regions close to the penetrating secondary jet flows); and
- IV low frequency fluctuations are in the regions with lower fluid velocities (e.g., inner core of the tank).

ACKNOWLEDGMENT

The authors would like to thank Bruce Power Inc. and Candu Owners Group (COG) for their financial and scientific support of this project.

REFERENCES

Austman, G., Szymanski, J., Garceau, M., Midvidy, W.I., "Measuring Moderator Temperature in a CANDU Reactor", 6th Annual Conference of CNS, Ottawa, Canada, 1985.

Carlucci, L.N., Cheung, I., "The Effects of Symmetric/Asymmetric Boundary Conditions on the Flow of an Internally Heated Fluid", Numerical Methods for Partial Differential Equations, 2, 1986, pp.4761~.

Hohne, T., Kliem, S., Vaibar, R., "Experimental and Numerical Modelling of Transition Matrix from Momentum to Buoyancy-Driven Flow in a Pressurized Water Reactor", Journal of Engineering for Gas Turbines and Power, 13, 2009.

Huget, R.G., Szymanski, J.K., Midvidy, W.I.,"Experimental and Numerical Modelling of Combined Forced and Free Convection in a Complex Geometry with Internal Heat Generation", Proceedings of 9th International Heat Transfer Conference, Jerusalem, Israel, 1990.

Huget, R.G., Szymanski, J.K., Midvidy, W.I., "Status of Physical and Numerical Modelling of CANDU Moderator Circulation", Proceedings of 10th Annual Conference of the Canadian Nuclear Society, 1989.

Khartabil, H. F., Inch, W. W., Szymanski, J., Novog, D., Tavasoli, V., Mackinnon, J., "Three-Dimensional Moderator Circulation Experimental Program for Validation of CFD code MODTURC_CLAS.", 21st CNS Nuclear Simulation Symposium, Ottawa, Canada, 2000.

Sion, N., "In-Core Moderator Temperature Measurement within CANDU Reactors", Nuclear Instruments and Methods, 206, 1983, 527-536.

Yoon, C., Rhee, B.W., Min, B.J., "3-D CFD Analysis of the CANDU-6 Moderator Circulation under Normal Operating Conditions", Journal of the Korean Nuclear Society, 36(6), 2004, 559- 570.

Yoon, C., Park, J.H., "Development of a CFD Model for the

CANDU-6 Moderator Analysis Using a Coupled Solver", Annals of Nuclear Energy, 35, 2008, pp.1041–1049.S. M. Metev and V. P. Veiko, Laser Assisted Microtechnology, 2nd ed., R. M. Osgood, Jr., Ed. Berlin, Germany: Springer-Verlag, 2005.

Araz Sarchami got his PhD from university of Toronto. He has extensive experience in numerical simulation field and currently he is working in nuclear industry focusing on the numerical simulation of reactor core.

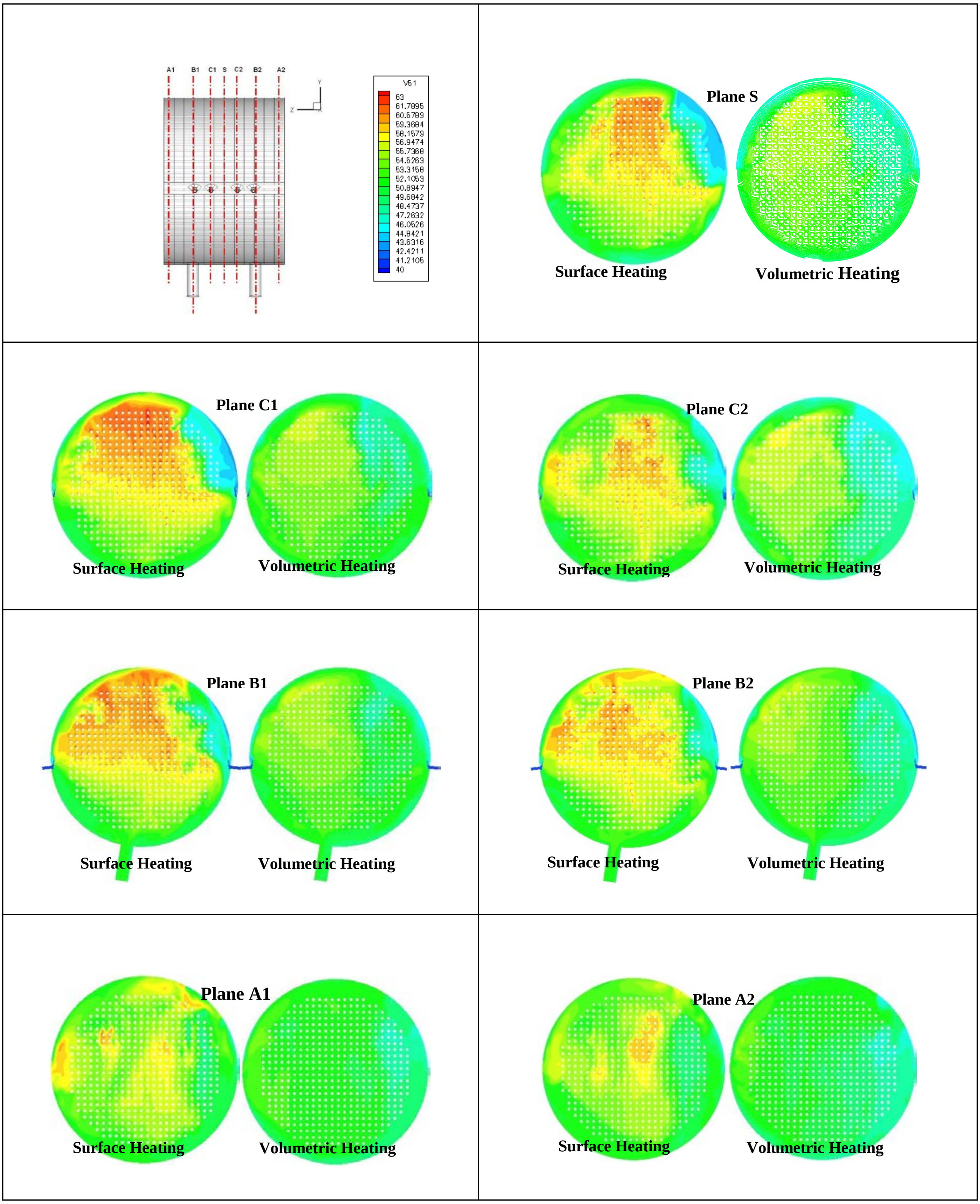

FIG. 4TEMPERATURE CONTOURS IN VARIOUS PLANES FOR SURFACE AND VOLUMETRIC HEATING CASES

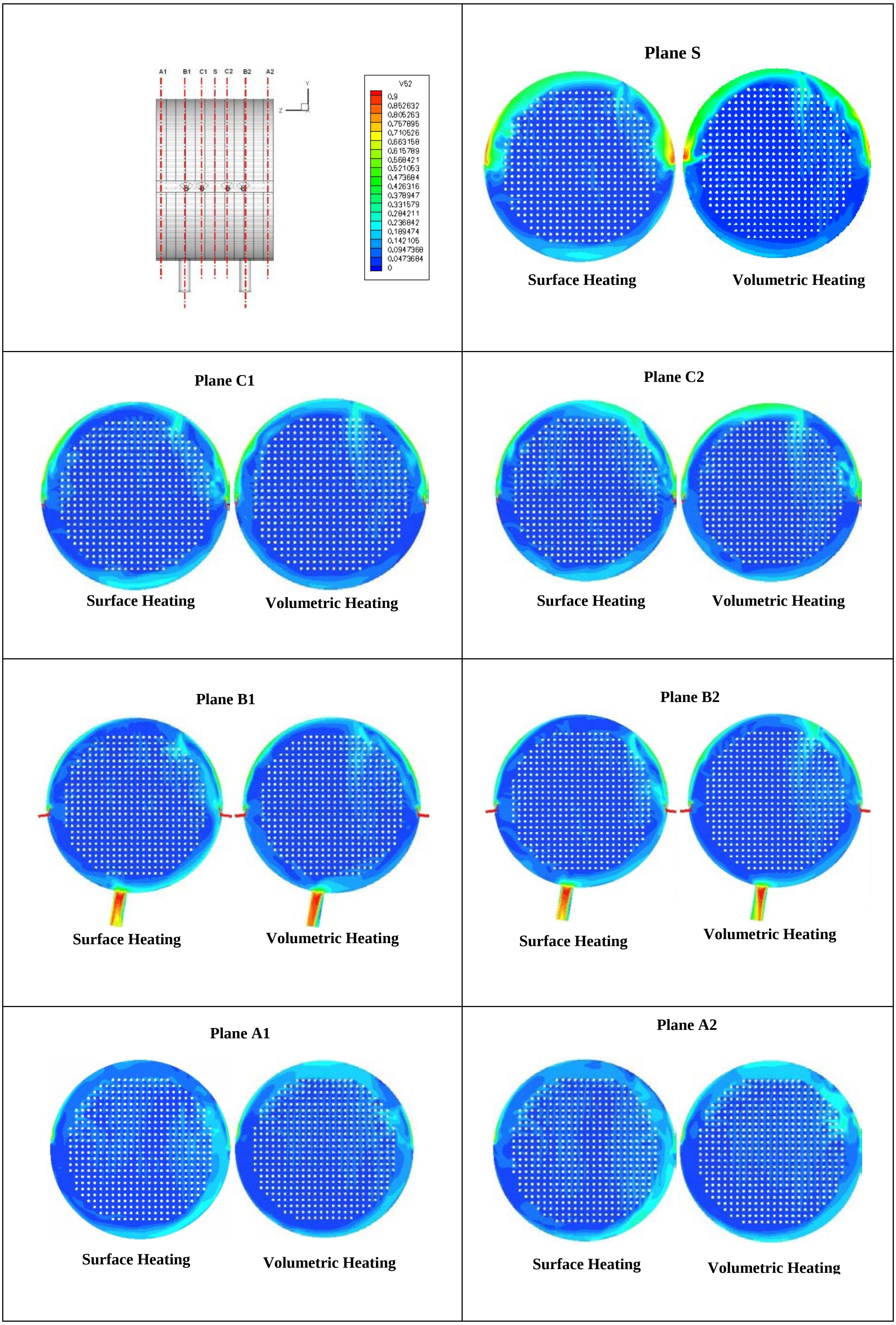

FIG. 5VELOCITY CONTOURS IN VARIOUS PLANES FOR SURFACE AND VOLUMETRIC HEATING CASES

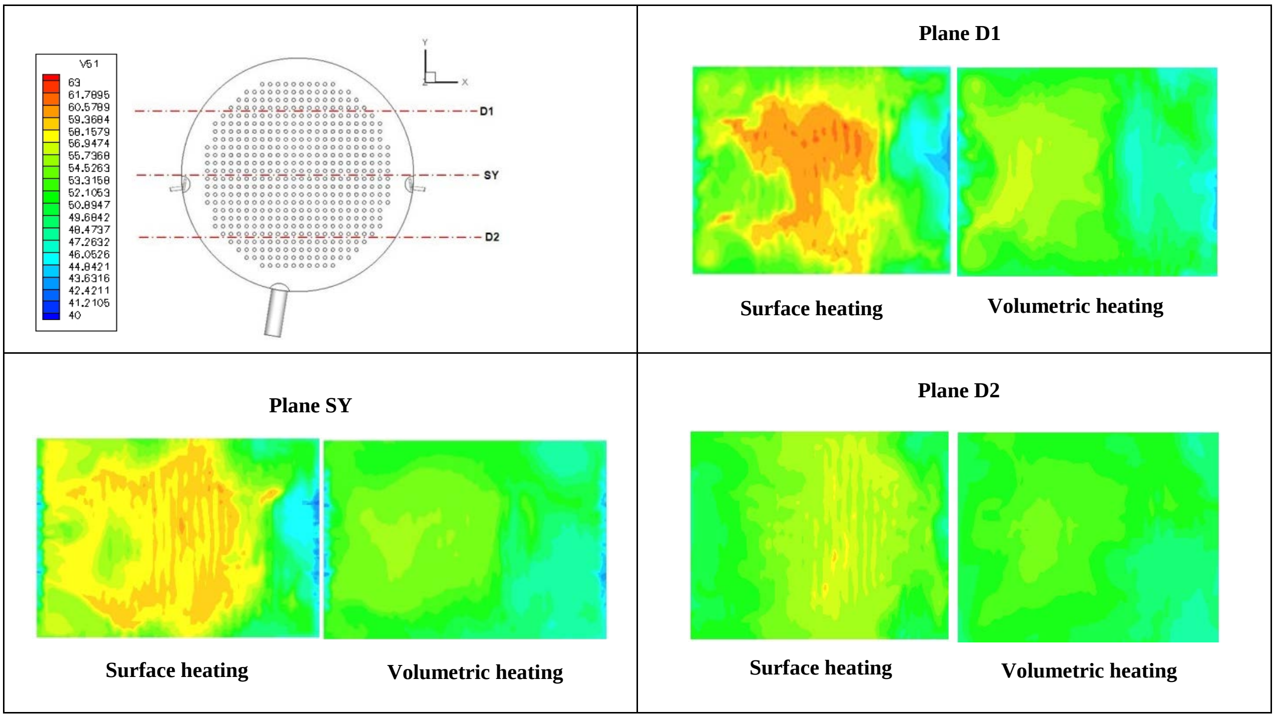

FIG. 6TEMPERATURE CONTOURS IN PLANES D1, SY, AND D2 FOR BOTH HEATING CASES

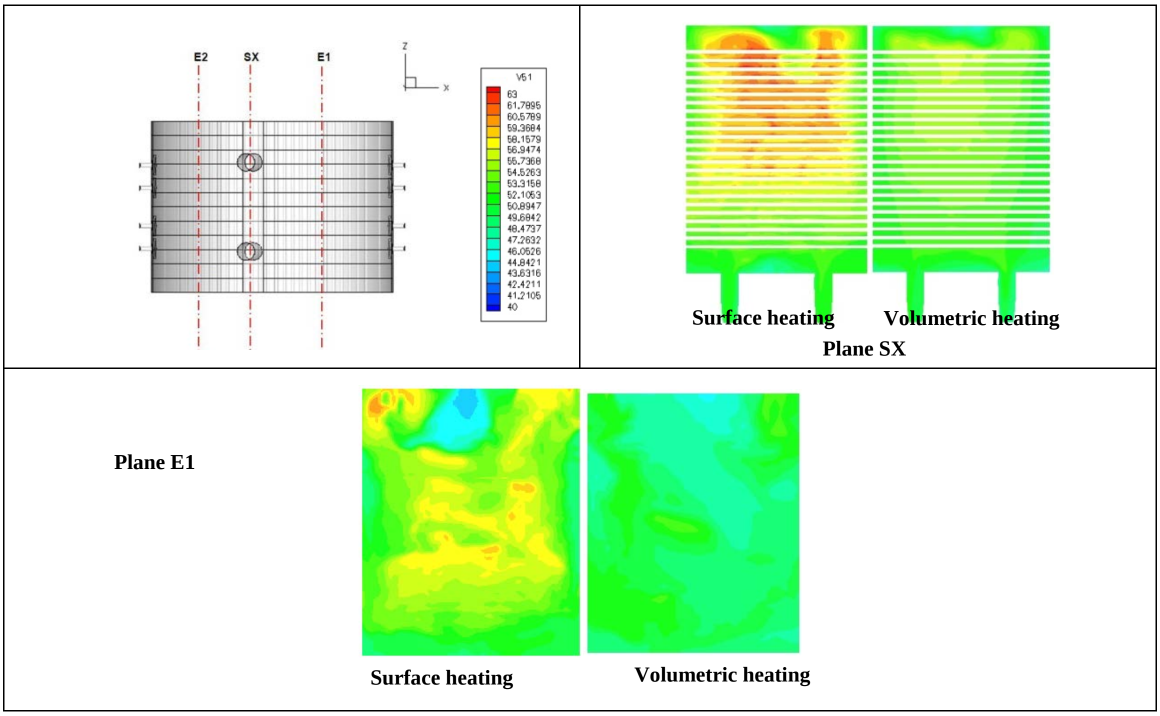

FIG. 7TEMPERATURE CONTOURS IN PLANES SX, AND E1 FOR BOTH HEATING CASES